

2010

Analysis of the Flow in Hermetic Compressor Valves Using the Immersed Boundary Method

Jonatas Ferreora Lacerda
Tecumseh do Brasil

Jose Luis Gasche
Universidade Estadual Paulista - Campus de Ilha Solteira

Follow this and additional works at: <https://docs.lib.purdue.edu/icec>

Lacerda, Jonatas Ferreora and Gasche, Jose Luis, "Analysis of the Flow in Hermetic Compressor Valves Using the Immersed Boundary Method" (2010). *International Compressor Engineering Conference*. Paper 1988.
<https://docs.lib.purdue.edu/icec/1988>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at <https://engineering.purdue.edu/Herrick/Events/orderlit.html>

Analysis of the Flow in Hermetic Compressor Valves Using the Immersed Boundary Method

Jônatas LACERDA^{1*}, José Luiz GASCHÉ²

^{1*}Tecumseh do Brasil LTDA, Research and Development Department,
Sao Carlos, Sao Paulo, Brazil
(55 16 3633 7678, jonatas.lacerda@tecumseh.com)

²UNESP – Ilha Solteira, Department of Mechanical Engineering,
Ilha Solteira, Sao Paulo, Brazil
(55 18 3743 1035, gasche@dem.feis.unesp.br)

* Corresponding Author

ABSTRACT

The flow in radial diffusers is numerically simulated in order to represent the flow through hermetic reciprocating compressors automatic valves. The Immersed Boundary method is used to represent solid boundaries inside the domain and the governing equations are solved using a numerical code based on the Finite Volume methodology. Initially, comparison between numerical and experimental results for pressure distribution on the reed, for two gaps distances and several Reynolds numbers, have shown that the methodology is suitable to study this problem. A chamfer was inserted in the inferior disk with three inclination angles, in order to show the flexibility of the method. As expected, the pressure gradient through the flow decreases for increasing inclination angles, indicating less amount of energy to drive the flow. Results showed that this is a promising methodology to study the flow through radial diffusers, with the future possibility of applying to real valve system of hermetic reciprocating compressors.

1. INTRODUCTION

In hermetic reciprocating compressors, the pressure difference between cylinder, suction and discharge chambers commands the movement of the valves. Once opened, the dynamic of the valves is controlled by force caused by the pressure field of the flow and the reaction force of the valves, which characterizes a fluid-structure interaction problem. Besides, complexes phenomena as flow separation, compressibility effects and turbulence occurring simultaneously during the flow through the valve turn the problem attractive to test the methodology.

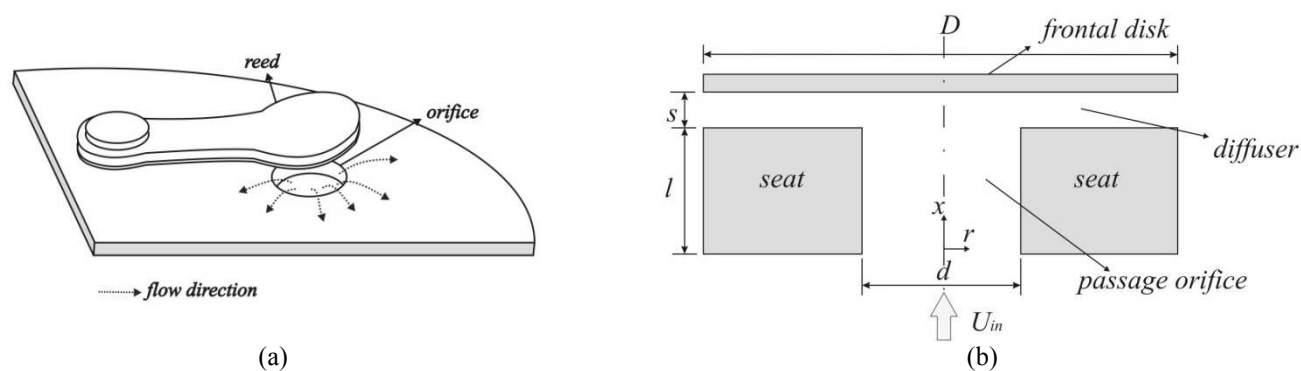


Figure 1: (a) Scheme of a valve reed and (b) radial diffuser geometry

Due to the complexity of this flow, simplified models have been adopted to study this problem, such as the radial diffuser geometry, showed in Figure 1. In this geometry, the fluid passes through an orifice and collides against the frontal disk (valve reed). The fluid is then deflected and flows between the reed and seat surfaces (diffuser), in direction to the outlet.

Ferreira and Driessen (1986) observed experimentally a great influence of the gap s/d on the flow configuration and the dimensionless pressure distribution on the reed. As shown in Figure 2, for small gaps ($s/d < 0.02$) there is no flow separation in the diffuser entrance, whereas, for higher gaps ($0.05 < s/d < 0.5$), it is observed the flow separation region that causes a great pressure drop. In real valves, this effect tends to close the valve, which is an undesired effect. Ferreira *et al.* (1989) observed experimentally and numerically that the presence of a chamfer in the diffuser entrance smoothes the pressure gradient and decreases the stagnation pressure. The authors also observed that a combination of higher values of Reynolds number and s/d can cause negative axial force that tends to close the valve.

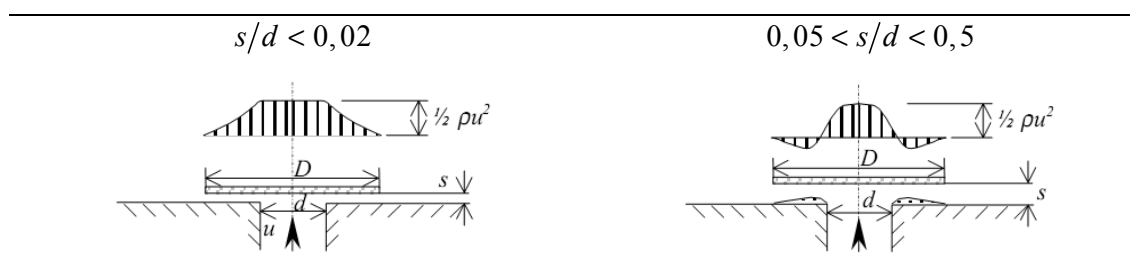


Figure 2: Dimensionless pressure distribution and flow configuration (adapted from Souto, 2002)

In order to represent more faithfully the valve geometry, some works have been performed in the last 20 years. Gasche (1992) analyzed experimentally and numerically the flow in eccentric radial diffusers, observing that the pressure and velocity fields are greatly modified due to the eccentricity, but there was no significant variation in the resulting force on the reed. Possamai *et al.* (2001) studied numerically the flow in radial diffuser with inclined reed, observing that small inclinations turn the pressure distribution considerably asymmetric.

Other authors have introduced the reed dynamics in the modeling. Lopes (1996) proposed a moving coordinate system, where the physical domain, which moves with the valve movement, is transformed in a numerical domain that remains fixed. The pressure distribution in the reed was used to determine the acting force on the reed, and an one-degree freedom model was adopted to determine its displacement. Matos *et al.* (2000, 2002 and 2006) improved the methodology developed by Lopes including turbulence, compressibility effects and the cylinder region, in order to represent more faithfully the compression cycle.

None of these works have treated this problem using the actual geometry including fluid-structure interaction, which is a difficult task. The Immersed Boundary (IB) method is a methodology that could be used to solve this complex problem without introducing the same difficulties encountered in other numerical methods. The goal of this work is to apply the IB method to study the flow through the radial diffuser, using a computational code based on the Finite Volume methodology, in order to solve the flow through the actual valve in the future.

2. THE IMMERSED BOUNDARY METHOD

The Immersed Boundary method, initially proposed by Peskin (1972), has been used successfully for studying fluid flow in complex geometries with or without fluid-structure interaction. The basic idea of this method is the use of a fixed Eulerian grid to solve the fluid flow problem together with a Lagrangian grid to represent body interfaces, which are also known as immersed boundaries. These two grids interact with each other by prescribing virtual forces in the momentum equations.

The advantage of the IB method is that the Lagrangian grid representing an interface can be displaced by the forces acting on it while the Eulerian grid can be always constructed using the Cartesian system and remains fixed. Therefore, the fluid flow problem can be solved without remeshing, which reduces the computational cost to solve

fluid-structure interaction problems. A comprehensive review about this subject can be found in Mittal and Iaccarino (2005).

2.1 Governing Equations

The two-dimensional, isothermal, unsteady and incompressible flow in a radial diffuser is governed by mass conservation, and Navier-Stokes equations, given, in cylindrical coordinate system, by:

Mass conservation:

$$\frac{\partial \rho}{\partial t} + \frac{1}{r} \frac{\partial(\rho r v)}{\partial r} + \frac{\partial(\rho u)}{\partial x} = 0 \quad (1)$$

Navier-Stokes in axial direction x :

$$\rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial r} + v \frac{\partial u}{\partial x} \right) = - \frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} + \frac{\partial^2 u}{\partial x^2} \right) + F_x \quad (2)$$

Navier-Stokes in radial direction r :

$$\rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial r} + v \frac{\partial v}{\partial x} \right) = - \frac{\partial p}{\partial r} + \mu \left(\frac{\partial^2 v}{\partial r^2} + \frac{1}{r} \frac{\partial v}{\partial r} - \frac{v}{r^2} + \frac{\partial^2 v}{\partial x^2} \right) + F_r \quad (3)$$

where u and v are the velocity components in x and r directions, respectively; p is the pressure; ρ is the density and μ is the dynamic viscosity. The terms F_x and F_r are the Eulerian force density components in x and r directions, respectively, and are responsible to represent an immersed boundary inside the flow, assuring the non-slip and impermeability conditions. There are several methods to calculate these forces. In this work, the Virtual Physical Model (VPM) proposed by Lima e Silva *et al.* (2004) is used to calculate F_x and F_r .

2.2 Virtual Physical Model (VPM)

In the IB method, an immersed boundary in the flow is defined by discrete Lagrangian points. The non-slip condition on a solid surface is prescribed by Eulerian forces in the Navier-Stokes equations (2) and (3), respectively. Before this step, the forces acting at each Lagrangian point must be determined and then distributed to the Eulerian domain. One manner to calculate this force has been proposed by Lima e Silva *et al.* (2003), and is known as Virtual Physical Model. In this model, the Lagrangian force density is obtained through a momentum balance in a fluid particle at each discrete Lagrangian point, \vec{x}_k , which gives the following equation:

$$\vec{f}(\vec{x}_k, t) = \underbrace{\rho \frac{\partial \vec{V}(\vec{x}_k, t)}{\partial t}}_{\vec{f}_a} + \underbrace{\rho(\vec{V} \cdot \vec{\nabla})\vec{V}(\vec{x}_k, t)}_{\vec{f}_i} - \underbrace{\mu \nabla^2 \vec{V}(\vec{x}_k, t)}_{\vec{f}_v} + \underbrace{\vec{\nabla} p(\vec{x}_k, t)}_{\vec{f}_p} \quad (4)$$

where \vec{f}_a , \vec{f}_i , \vec{f}_v e \vec{f}_p represent, respectively, the acceleration, inertial, viscous and pressure forces (by volume unit) acting on the fluid particle at the interface.

All terms described in equation (4) must be calculated at the interface using the velocity field, $\vec{V}(\vec{x})$, and the pressure field, $p(\vec{x})$, obtained by proper interpolations from the Eulerian domain. In order to assure the non-slip and impermeability conditions, the flow velocity at the interface must be equal to the interface velocity.

The acceleration term \overline{fa} is calculated by $\rho(\vec{V}_k - \vec{V}_{fk})/\Delta t$, where \vec{V}_k and \vec{V}_{fk} are the interface velocity and the fluid velocity at the interface, respectively, for the Lagrangian point \vec{x}_k . The calculation of the spatial derivatives in the other terms of the Lagrangian force density are performed by Lagrange polynomials. Four auxiliary points are defined as illustrated in Figure 3 and the values of pressure and velocities at these points are interpolated from the Eulerian domain by the following distribution function:

$$\phi(\vec{x}_k) = \sum_i \sum_j D_{ij}(\|\vec{x}_{ij} - \vec{x}_k\|) \phi(\vec{x}_{ij}) \quad (5)$$

where ϕ represents the pressure or the velocity components, \vec{x}_k represents the Lagrangian and auxiliary points, \vec{x}_{ij} represents a volume in Eulerian domain, and D_{ij} is given by:

$$D_{ij}(\|\vec{x}_{ij} - \vec{x}_k\|) = g(m_x) \cdot g(m_r) = g\left(\frac{\|x_{ij} - x_k\|}{\Delta x}\right) \cdot g\left(\frac{\|r_{ij} - r_k\|}{\Delta r}\right) \quad (6)$$

where Δx and Δr are the dimensions of the Eulerian control volume in the x and r directions, respectively. The values of $g(m_x)$ and $g(m_r)$, are calculated by substituting m by m_x or m_r in the following function:

$$g(m) = \begin{cases} \frac{3 - 2\|m\| + \sqrt{1 + 4\|m\| - 4\|m\|^2}}{8}, & \text{se } m < 1 \\ \frac{1}{2} - \frac{3 - 2\|2 - m\| + \sqrt{1 + 4\|2 - m\| - 4\|2 - m\|^2}}{8}, & \text{se } 1 < m < 2 \\ 0, & \text{se } m > 2 \end{cases} \quad (7)$$

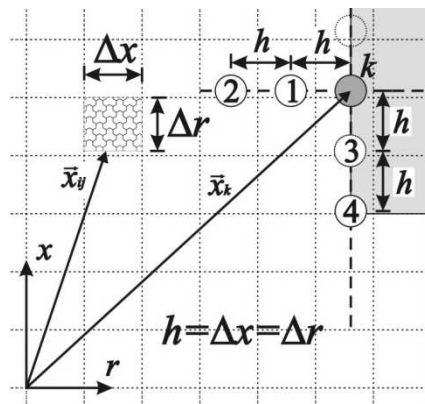


Figure 3: Lagrangian (k) and auxiliary (1, 2, 3 and 4) points

The derivatives are calculated by using a second order Lagrange polynomial through the points ϕ_1 , ϕ_2 , ϕ_3 and ϕ_4 , which are previously interpolated values. Finally, the Eulerian density force field, $F_i(\vec{x}_{ij})$, is obtained from the Lagrangian density force, $f_i(\vec{x}_k)$, by:

$$F_i(\bar{x}_{ij}) = \sum_k D_{ij}(\|\bar{x}_{ij} - \bar{x}_k\|) f_i(\bar{x}_k) \frac{\Delta V_k}{\Delta V_{ij}} \quad (8)$$

where D_{ij} is the interpolation/distribution function defined in equation (6), and $\Delta V_k / \Delta V_{ij}$ is the ratio between the Lagrangian volume and Eulerian volume.

The immersed boundaries in the Eulerian domain are virtually modeled by equations (2) and (3) with the density force field F_i . The calculation of F_i is repeated at each time step, assuming that the mass conservation is reached and the non-slip and impermeability conditions are obtained at the interface, which are verified by the norm L_2 , defined by:

$$L_2 = \frac{\sqrt{\left[(u_{fk} - u_k)^2 + (v_{fk} - v_k)^2 \right]}}{N} \quad (9)$$

where N is the number of Lagrangian points used to represent the interface. Values of the order of 10^{-2} for L_2 are generally considered acceptable.

3. NUMERICAL SIMULATION

The geometry of the radial diffuser and boundary conditions used in the numerical simulation are shown in the Figure 4, where u is the velocity component in the x -direction and v is the velocity component in the r -direction. A constant velocity, U_{in} , is used at the inlet of the orifice. In this work, the seat region was represented by the IB method through two groups of Lagrangian points.

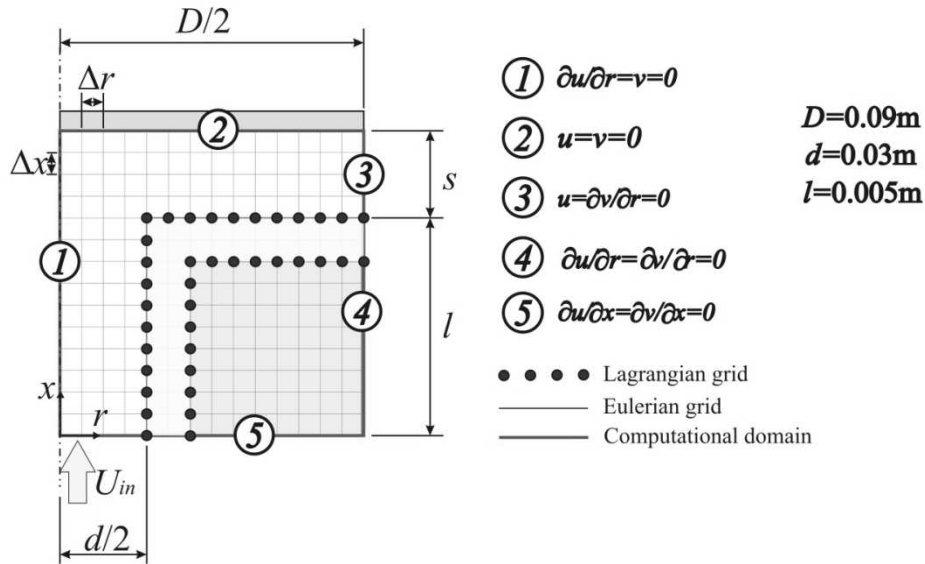


Figure 4: Radial diffuser computational domain and boundary conditions

This problem was also studied numerically and experimentally by Gasche (1992), who obtained the dimensionless pressure distribution on the reed, $p_{adm} = p / (0.5 \cdot \rho U_{in}^2)$, for several Reynolds numbers, $Re = \rho U_{in} d / \mu$, and different gaps between disks. These results are used in the present work to confront with the numerical results obtained from the IB method.

4. RESULTS

4.1 Comparison with experimental data

Figure 5 shows the comparison between the experimental data from Gasche (1992) and the numerical results for the dimensionless pressure on the reed. For Reynolds numbers below 2000 was used Central-Difference (CDS) as interpolation scheme for advective terms, and Power-law interpolation scheme for Reynolds above 2000, due to convergence difficulty encountered for higher Reynolds numbers.

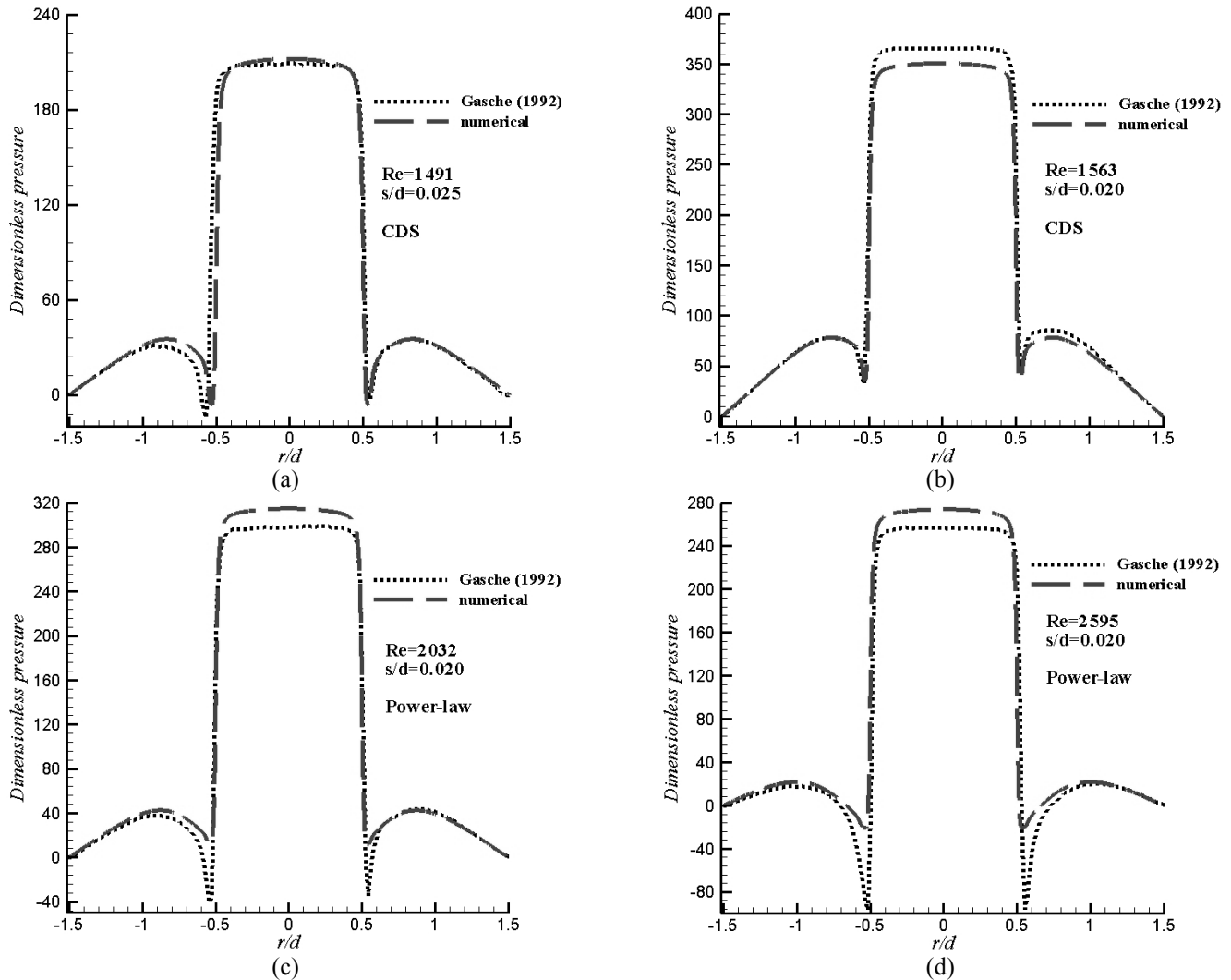


Figure 5: Experimental and numerical results for dimensionless pressure on the reed

The pressure distributions show a stagnation region corresponding to the diameter of the orifice ($r/d < 0.5$) where the fluid velocity is close to zero. At the entrance of the diffuser ($r/d \sim 0.5$) there is a high pressure drop due to the acceleration of the flow. The pressure can reach negative values for high Reynolds numbers, as observed in Figures 5-(c) and 5-(d). As Power-law scheme presents high numerical diffusion, at the inlet of the diffuser, the numerical pressure drop is lower than the experimental result. The CDS used in cases (a) and (b) produced very good agreement between numerical and experimental data. In general, the numerical results showed good agreement with experimental data, demonstrating that this methodology is suitable to study this problem.

4.2 Geometrical analysis of seat

An advantage of IB method is the flexibility to modify easily the geometry. In order to show that it was studied the effect of a chamfer at the entrance of the diffuser on the dimensionless pressure distribution on the reed. Figure 6 shows the geometry of the chamfer and Figures 7 and 8 depict the dimensionless pressure for several inclination angles ($\alpha = 0^\circ, 30^\circ, 45^\circ$ and 60°), considering $e/s=1$.

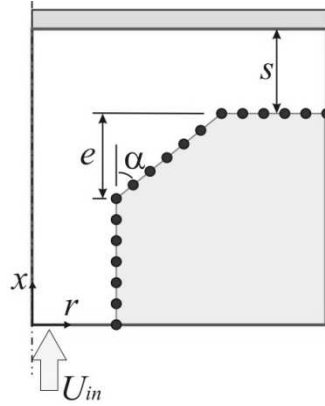
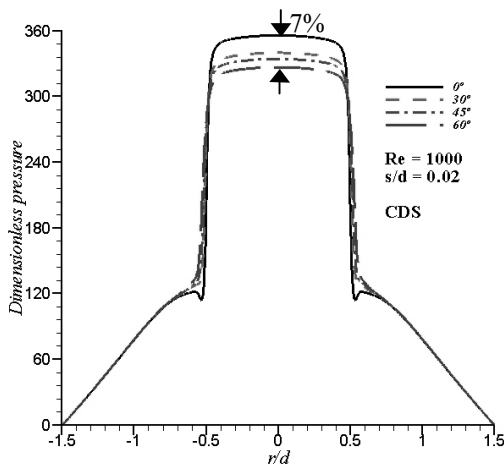
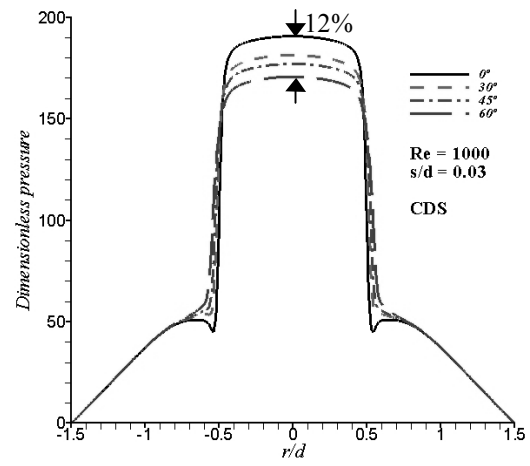


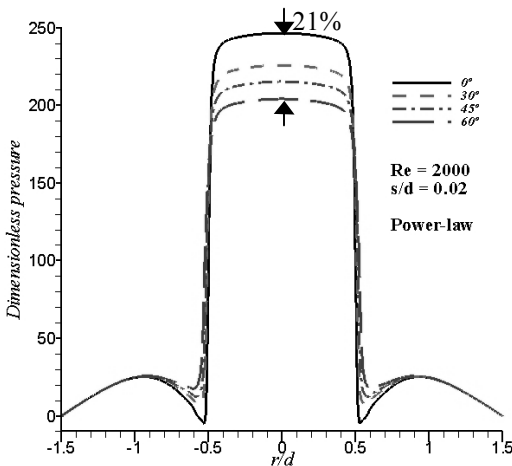
Figure 7: Radial diffuser with a chamfer



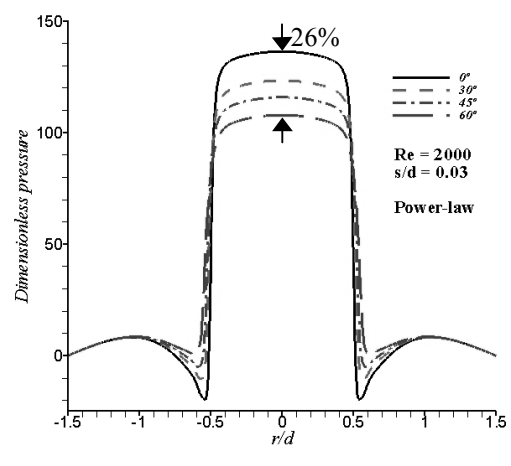
(a)



(b)



(c)



(d)

Figure 8: Dimensionless pressure distribution for varying Reynolds number and relation s/d

Observing Figures 8 and 9 it is possible to notice that the total pressure drop of the flow decreases for increasing chamfer angles, as expected. This is so because the reduction of the local pressure drop at the entrance of the diffuser due to the presence of the chamfer, which enlarges the cross section area, diminishing the curvature of the flow. This effect is desirable for the compressor operation, in order to reduce the consumption of energy.

6. CONCLUSIONS

The flow in radial diffuser representing the valve system of hermetic compressors is numerically solved using the Immersed Boundary method and the Virtual Physical Model to calculate the forces acting on the boundaries. The numerical methodology was validated by experimental data, showing good agreement. In order to show the flexibility of the method to represent easily complex geometries, a chamfer was introduced at the entrance of the diffuser region, which reduces the total pressure drop of the flow. The main conclusion of the work is that the IB method is a promising methodology to predict the actual behavior of the valve, considering the fluid-structure interaction problem.

NOMENCLATURE

d	orifice diameter	(m)	e	chamfer height	(m)
D	disk diameter (reed)	(m)	p_{adm}	dimensionless pressure	(–)
s	gap distance	(m)	Re	Reynolds number	(–)
l	seat length	(m)	α	chamfer angle	(°)

REFERENCES

- Ferreira, R. T. S., Driessen, J. L., 1986, Analysis of the influence of valve geometric parameters on the effective flow and force areas, *International Compressor Engineering Conference at Purdue*, p. 632-646.
- Ferreira, R. T. S., Prata A. T., Deschamps C. J., 1989, Pressure distribution along valve reeds of hermetic compressors, *Experimental Thermal and Fluid Science*, vol. 2, no. 2: p. 201-207.
- Gasche, J. L., 1992, Laminar flow through excentric valves of refrigeration compressors, MS. Dissertation, (in Portuguese), Federal University of Santa Catarina, Florianópolis-SC, Brazil, 149 p.
- Lima e Silva, A. L. F., Silveira-Neto, A., Damasceno, J. J. R., 2004, Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method, *Journal of Computational Physics*, vol. 189, no. 2: p. 351-370.
- Lopes, M. N., 1996, A numerical methodology for analysis of the dynamic behavior of reed type valves in periodic flows, MS. Dissertation, (in Portuguese), Federal University of Santa Catarina, Florianópolis-SC, Brazil, 94 p.
- Matos, F. F. S., Prata, A. T., Deschamps, C. J., 2000, A numerical methodology for the analysis of valve dynamics, *International Compressor Engineering Conference at Purdue*, p. 383-390.
- Matos, F. F. S., Prata, A. T., Deschamps, C. J., 2002, Numerical simulation of the dynamics of reed type valves, *International Compressor Engineering Conference at Purdue*, C15-2, 8 p.
- Matos, F. F. S., Prata, A. T., Deschamps, C. J., 2006, A two-dimensional simulation model for reciprocating compressors with automatic valves, *International Compressor Engineering Conference at Purdue*, C053, 8 p.
- Mittal, R., Iaccarino, G., 2005, Immersed boundary methods, *Annual Review of Fluid Mechanics*, vol. 37: p. 239-261.
- Peskin, C. S., 1972, Flow pattern around heart valves: a numerical method, *Journal of Computational Physics*, vol. 10, no. 2: p. 252-271.
- Possamai, F. C., Ferreira R. T. S., Prata, A. T., 2001, Pressure distribution in laminar radial flow through inclined disks, *International Journal of Heat Fluid Flow*, vol. 22, no. 4: p. 440-449.
- Souto, L. E. M., 2002, Experimental investigation of turbulent flow in compressors, MS. Dissertation, (in Portuguese), Federal University of Santa Catarina, Florianópolis-SC, Brazil, 105 p.

ACKNOWLEDGEMENT

To Fundação de Amparo à Pesquisa do Estado de São Paulo (FAPESP) for the financial support.